

REPORT DOCUMENTATION PAGE				Form Approved OMB No. 0704-0188	
The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing the burden, to the Department of Defense, Executive Service Directorate (0704-0188). Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.					
PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ORGANIZATION.					
1. REPORT DATE (DD-MM-YYYY) 11-06-2010		2. REPORT TYPE Final		3. DATES COVERED (From - To) January 2007 - June 2010	
4. TITLE AND SUBTITLE High-Order Numerical Algorithms for Steady and Unsteady Simulation of Viscous Compressible Flow with Shocks (Grant FA9550-07-1-0195) - Final Report				5a. CONTRACT NUMBER	
				5b. GRANT NUMBER FA9550-07-1-0195	
				5c. PROGRAM ELEMENT NUMBER	
				5d. PROJECT NUMBER	
6. AUTHOR(S) Jameson, Antony Vincent, Peter E				5e. TASK NUMBER	
				5f. WORK UNIT NUMBER	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Stanford Universit, Stanford, California, USA, 94305				8. PERFORMING ORGANIZATION REPORT NUMBER	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) Air Force Office of Scientific Research				10. SPONSOR/MONITOR'S ACRONYM(S) AFOSR	
				11. SPONSOR/MONITOR'S REPORT NUMBER(S) AFRL-OSR-VA-TR-2012-0179	
12. DISTRIBUTION/AVAILABILITY STATEMENT  DISTRIBUTION A: APPROVED FOR PUBLIC RELEASE					
13. SUPPLEMENTARY NOTES					
14. ABSTRACT In the following document a summary is given of research carried out under award number FA9550-07-1-0195 from the Air Force Office of Scientific Research. The objective of the proposed research was to develop and implement high-order numerical algorithms for the simulation of steady and unsteady compressible viscous flows with shocks. Notable achievements have been made in three areas, namely algorithm analysis and development, code development, and code utilization (to investigate various flow problems). With regards to code development, two-dimensional and three-dimensional compressible viscous flow solvers based on the spectral difference method have been written. The solvers can run on meshes containing straight-sided and curved-sided quadrilateral and hexahedral elements. With regards to code utilization, viscous compressible flow over various two-dimensional configurations has been investigated. These configurations include pairs of cylinders (both stationary and rotating), plunging and pitching airfoils, and a deforming beam in the wake of a cylinder. In three-dimensions, turbulent channel flow and turbulent flow over an airfoil have been investigated, and preliminary simulations of viscous compressible flow over a flapping wing have been undertaken.					
15. SUBJECT TERMS					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES  32	19a. NAME OF RESPONSIBLE PERSON Jameson, Antony
a. REPORT	b. ABSTRACT	c. THIS PAGE			19b. TELEPHONE NUMBER (Include area code) (650) 725-6208

**High-Order Numerical Algorithms for  
Steady and Unsteady Simulation of  
Viscous Compressible Flow with Shocks  
(Grant FA9550-07-1-0195)**

**Final Report**

Principal Investigator: Antony Jameson

June 11, 2010

*Aerospace Computing Laboratory, Department of Aeronautics and Astronautics,  
Stanford University, Stanford, California, 94305*

## **Abstract**

In the following document a summary is given of research carried out under award number FA9550-07-1-0195 from the Air Force Office of Scientific Research. The objective of the proposed research was to develop and implement high-order numerical algorithms for the simulation of steady and unsteady compressible viscous flows with shocks. Notable achievements have been made in three areas, namely algorithm analysis and development, code development, and code utilization (to investigate various flow problems). With regards to algorithm analysis and development, it has been proved for one-dimensional linear advection that the spectral difference method is stable for all orders of accuracy in a norm of Sobolev type (provided that the interior flux collocation points are placed at the zeros of the corresponding Legendre polynomials). Also, a new range of energy stable high-order methods based on the so called flux reconstruction approach have been identified. With regards to code development, two-dimensional and three-dimensional compressible viscous flow solvers based on the spectral difference method have been written. The solvers can run on meshes containing straight-sided and curved-sided quadrilateral and hexahedral elements. Within the aforementioned codes a range of shock capturing, automatic mesh refinement, mesh deformation, and convergence acceleration algorithms have been implemented and tested. With regards to code utilization, viscous compressible flow over various two-dimensional configurations has been investigated. These configurations include pairs of cylinders (both stationary and rotating), plunging and pitching airfoils, and a deforming beam in the wake of a cylinder. In three-dimensions, turbulent channel flow and turbulent flow over an airfoil have been investigated, and preliminary simulations of

viscous compressible flow over a flapping wing have been undertaken.

Keywords: *High-Order Methods, Spectral Difference Methods, Shock Capturing, Deforming Meshes, Time Integration Schemes*

## Contents

<b>1</b>	<b>Introduction</b>	<b>4</b>
<b>2</b>	<b>Accomplishments</b>	<b>6</b>
2.1	Overview . . . . .	6
2.2	Algorithm Analysis and Development . . . . .	10
2.2.1	Proof of Stability of the Spectral Difference Method . . . . .	10
2.2.2	Development of Energy Stable Flux Reconstruction Schemes . . . . .	11
2.3	Code Development . . . . .	11
2.3.1	Two-Dimensional Compressible Viscous Flow Solver . . . . .	11
2.3.2	Three-Dimensional Compressible Viscous Flow Solver . . . . .	12
2.3.3	Shock Capturing . . . . .	12
2.3.4	Automatic Mesh Refinement . . . . .	14
2.3.5	Mesh Deformation . . . . .	16

2.3.6	Convergence Acceleration . . . . .	18
2.4	Code Utilization . . . . .	19
2.4.1	Two-Dimensional Flow over Cylinders . . . . .	19
2.4.2	Two-Dimensional Plunging and Pitching Airfoils . . . . .	20
2.4.3	Two-Dimensional Interaction of a Cylinder Wake with a De- forming Beam . . . . .	22
2.4.4	Three-Dimensional Turbulent Channel Flow . . . . .	24
2.4.5	Three-Dimensional Turbulent Flow Over an Airfoil . . . . .	24
2.4.6	Three-Dimensional Flapping Wing . . . . .	26
<b>3</b>	<b>Conclusions</b>	<b>27</b>

# 1 Introduction

High-order numerical methods potentially offer better accuracy than low-order schemes for a comparable computational cost. However, existing high-order methods are generally less robust and more complex to implement than their low-order counterparts. These issues, in conjunction with difficulties generating high-order meshes, have prevented the wide-spread adoption of high-order techniques in either academia (where the use of low-order schemes remains widespread) or in industry (where the use of low-order schemes is ubiquitous).

The most mature and widely used high-order methods (at least for unstructured grids) are based on a class of schemes developed in 1973 by Reed and Hill [1] to solve the neutron transport equation. Such schemes have become known as discontinuous Galerkin (DG) methods, and numerous variants have been developed for solving the weak form of both hyperbolic [2] and elliptic systems [3]. The basic principle of DG schemes is to decompose the approximate numerical solution both spatially, by tessellating a given computational domain with separate elements, and also spectrally, via a summation of piecewise discontinuous polynomial basis functions within each element. A particularly simple and efficient range of DG schemes utilize high-order Lagrange polynomial basis functions inside each element, defined by solution values at a set of distinct nodal points. Such schemes have become known as nodal DG methods, an exposition of which can be found in the recent textbook by Hesthaven and Warburton [4], as well as in various articles by the same authors [5][6]. Similar to nodal DG methods are spectral difference (SD) methods, (although unlike nodal DG methods, SD methods are based on the governing system in its differential form). The foundation for such schemes was first put forward by Kopriva and Kolas [7] in 1996 under the name of “staggered grid Chebyshev multidomain” methods. However, several years later in 2006 Liu, Wang and Vinokular [8] presented a more general formulation for both triangular and quadrilateral elements, which they termed the SD method (a name which has been retained to the present).

In the study presented here, a broad effort to investigate, implement, test and ultimately improve the SD method was undertaken, with particular emphasis placed on issues such as shock capturing and efficient time integration, which have hitherto inhibited the widespread adoption of high-order methods amongst a wider scientific

community.

## 2 Accomplishments

### 2.1 Overview

With regards to algorithm analysis and development, the main accomplishments of the research can be summarized as follows:

- A proof demonstrating that particular SD schemes are stable (for linear advection) [9].
- The development of a new range of energy stable flux reconstruction (FR) schemes [10].

With regards to code development, the main accomplishments of the research can be summarized as follows:

- The development of a two-dimensional (2D) viscous compressible SD flow solver.
- The development of a three-dimensional (3D) viscous compressible SD flow solver.
- The implementation of shock capturing algorithms suitable for use with the SD method [11][12].

- The implementation of automatic mesh refinement algorithms suitable for use with the SD method [12].
- The implementation of mesh deformation algorithms suitable for use with the SD method [13].
- The implementation of convergence acceleration schemes suitable for use with the SD method [14].

With regards to code utilization, the main accomplishments of the research can be summarized as follows:

- Studies of 2D viscous compressible flow over pairs of cylinders (both stationary and rotating) [15][16].
- Studies of 2D viscous compressible flow over pitching and plunging airfoils [13].
- Studies of 2D viscous compressible flow over a deforming beam in a cylinder wake.
- Studies of 3D turbulent channel flow [17].
- Studies of 3D turbulent flow over airfoils.
- Preliminary studies of 3D flow over a flapping wing.

Journal articles resulting from the research include:

- P. E. Vincent, P. Castonguay, A. Jameson. A New Class of High-Order Energy Stable Flux Reconstruction Schemes. Submitted to Journal of Scientific Computing. May 2010.
- G. May, F. Iacono, A. Jameson. A Hybrid Multilevel Method for High-Order Discretization of the Euler Equations on Unstructured Meshes, Journal of Computational Physics. May 2010.
- A. Jameson. A Proof of the Stability of the Spectral Difference Method for All Orders of Accuracy, Journal of Scientific Computing. January 2010.
- C. Liang, S. Premasuthan, A. Jameson. High-order Accurate Simulation of Low-Mach Laminar Flow Past Two Side-by-Side Cylinders Using Spectral Difference Method. Journal of Computers and Structures. February 2009.
- C. Liang, A. Jameson, Z. Wang. Spectral Difference Method for Compressible Flow on Unstructured Grids with Mixed Elements, Journal of Computational Physics. January 2009.

Full length conference papers resulting from the research include:

- P. Castonguay, A. Jameson. Simulation of Transitional Flow over Airfoils using the Spectral Difference Method. To be presented at the AIAA Computational Fluid Dynamics Meeting. Chicago, Illinois. June 2010.
- K. Ou, A. Jameson. A High-Order Spectral Difference Method for Fluid-Structure Interaction on Dynamic Deforming Meshes. To be presented at the AIAA Computational Fluid Dynamics Meeting. Chicago, Illinois. June 2010.

- Y. Li, S. Premasuthan, A. Jameson. Comparison of h and p Adaptations for Spectral Difference Methods. To be presented at the AIAA Computational Fluid Dynamics Meeting. Chicago, Illinois. June 2010.
- S. Premasuthan, C. Liang, A. Jameson. Computation of Flows with Shocks Using Spectral Difference Scheme with Artificial Viscosity, AIAA Aerospace Sciences Meeting. Orlando, Florida. January 2010.
- K. Ou, C. Liang, A. Jameson. A High-Order Spectral Difference Method for the Navier-Stokes Equations on Unstructured Moving Deformable Grids, AIAA Aerospace Sciences Meeting. Orlando, Florida. January 2010.
- C. Liang, S. Premasuthan, A. Jameson, Z. Wang. Large Eddy Simulation of Compressible Turbulent Channel Flow with Spectral Difference Method. AIAA Aerospace Sciences Meeting. Orlando, Florida. January 2009.
- S. Premasuthan, C. Liang, A. Jameson, Z. Wang. p-Multigrid Spectral Difference Method For Viscous Compressible Flow Using 2D Quadrilateral Meshes. AIAA Aerospace Sciences Meeting. Orlando, Florida. January 2009.
- S. Premasuthan, C. Liang, A. Jameson. A Spectral Difference Method for Viscous Compressible Flows With Shocks. AIAA Computational Fluid Dynamics Meeting. San Antonio, Texas. June 2009.
- K. Ou, C. Liang, S. Premasuthan, A. Jameson. High-Order Spectral Difference Simulation of Laminar Compressible Flow Over Two Counter-Rotating Cylinders. AIAA Computational Fluid Dynamics Meeting. San Antonio, Texas. June 2009.

PhD dissertations resulting from the research include:

- S. Premasuthan. Towards an Efficient and Robust High Order Accurate Flow Solver for Viscous Compressible Flow. Ph.D. Dissertation, Stanford University, March 2010.

Awards received during the research period include:

- A. Jameson. Elmer A. Sperry Award.

## 2.2 Algorithm Analysis and Development

### 2.2.1 Proof of Stability of the Spectral Difference Method

It has been demonstrated that for the case of one-dimensional (1D) linear advection, the SD method is stable for all orders of accuracy in a norm of Sobolev type (provided that the interior flux collocation points are placed at the zeros of the corresponding Legendre polynomials). The proof is based on an energy method. For solution polynomials of degree  $k$  (which result in a scheme of order  $k+1$ ), stability is demonstrated with a norm of the form

$$||u|| = \int (u^2 + \beta^{2k} c u^{(k)^2}) dx, \quad (2.1)$$

where  $\beta$  is a piecewise constant scaling factor and  $c$  is a  $k$  dependent coefficient. For further details see Jameson [9].

### **2.2.2 Development of Energy Stable Flux Reconstruction Schemes**

The FR approach [18] to high-order methods is robust, efficient, simple to implement, and allows various high-order schemes, such as the nodal DG method and the SD method, to be cast within a single unifying framework. We have identified a new class of one-dimensional energy stable FR schemes. The energy stable schemes are parameterized by a single scalar quantity, which if chosen judiciously leads to the recovery of various well known high-order methods (including the nodal DG method and a particular SD method), as well as one other FR scheme that was previously found to be stable by Huynh [18]. The analysis offers significant insight into why certain FR schemes are stable, whereas others are not. Also, from a practical standpoint, the analysis provides a simple prescription for implementing an infinite range of energy stable high-order methods via the particularly intuitive FR approach. We are currently working to extend the formulation to simplex elements. For further details see Vincent, Castonguay and Jameson [10].

## **2.3 Code Development**

### **2.3.1 Two-Dimensional Compressible Viscous Flow Solver**

A 2D viscous compressible SD flow solver has been written in FORTRAN. The solver, which is based on the approach presented by Wang [19], can run on meshes containing both straight-sided and curved-sided quadrilateral elements. An methodology for applying the code to unstructured mixed meshes containing both triangular and quadrangular elements has also been developed [20].

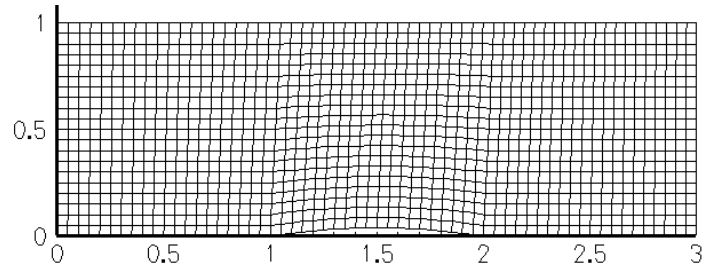
### 2.3.2 Three-Dimensional Compressible Viscous Flow Solver

A 3D viscous compressible SD flow solver has been written in FORTRAN. The solver, which is based on the approach presented by Wang [19], can run on meshes containing both straight-sided and curved-sided hexahedral elements. To facilitate simulating flow in large 3D geometries the solver has been parallelized for use on CPU clusters using MPI.

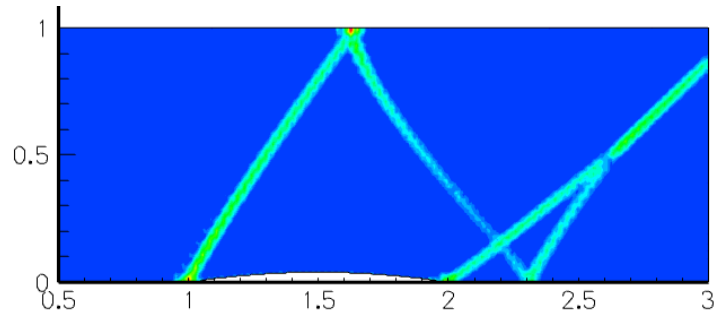
### 2.3.3 Shock Capturing

One of the greatest restrictions of high-order unstructured solvers is their inability to handle flow discontinuities. When flows develop steep gradients such as shock waves or contact surfaces, non-physical spurious oscillations arise that cause the simulations to become unstable. For higher-order approximations, it is typically necessary to add explicit dissipation in order to obtain a stable solution. But this has a negative effect on accuracy, and the resolution of turbulent scales. The development of numerical algorithms that capture discontinuities and also resolve the scales of turbulence in compressible turbulent flows remains a significant challenge.

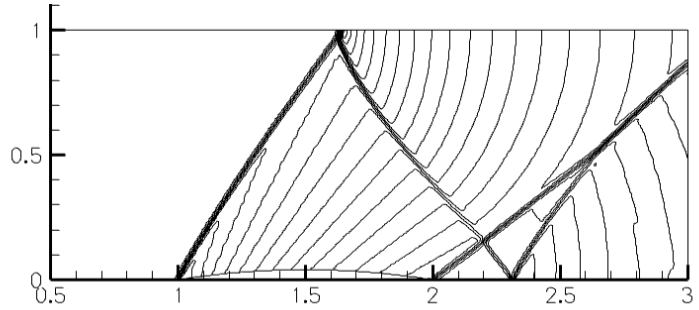
In this study the shock capturing approach of Cook and Cabot [21] has been adapted to the SD method. The scheme adds artificial viscosity to the flow when discontinuities are present [11][12]. This causes the discontinuities to be smeared out over a single (high-order) SD element, thus precluding the development of any spurious oscillations. Results illustrating the shock pattern for supersonic 2D flow over a bump are shown in Fig. 1.



(a)



(b)



(c)

Figure 1: Shock pattern resulting from 2D supersonic flow over a bump calculated using a third-order SD method with artificial viscosity. The computational mesh is shown in (a), artificial bulk viscosity is shown in (b) and pressure contours are shown in (c). Note that artificial bulk viscosity is only added in the presence of shocks.

### 2.3.4 Automatic Mesh Refinement

An optimal computational mesh should be able to capture details of the flow solution, yet avoid unnecessary resolution of regions in which the flow is predominantly uniform. In order to generate such optimal meshes *a priori* knowledge of the flow physics is required, as well as an understanding of how the chosen numerical scheme performs. Obtaining such information, and generating such optimal meshes, is often very difficult, if not impossible. Therefore, to mitigate these issues, various adaptive meshing techniques have been developed.

Adaptive meshing techniques allow the mesh to be automatically modified based on the flow solution. The nature of these modifications is usually determined by error indicators calculated from the flow as it develops (for example local entropy error may be used as an indicator for isentropic flows). Regions with higher error are then refined, and in some cases regions with lower errors unrefined. For high-order schemes, such as SD and DG type methods, refinement can occur in a variety of ways. These include reduction of the element size (*h*-type refinement), increasing the element order (*p*-type refinement), or a combination of both (*hp*-type refinement).

Various error estimation and mesh refinement strategies have been implemented and tested within our in-house 2D SD flow solver [12]. An illustration of how *h*-type refinement has been used to improve resolution of shock waves (forming due to supersonic flow over a bump) is shown in Fig. 2. Note that artificial viscosity is also employed to ensure that the SD scheme is stable in the presence of the shock waves.

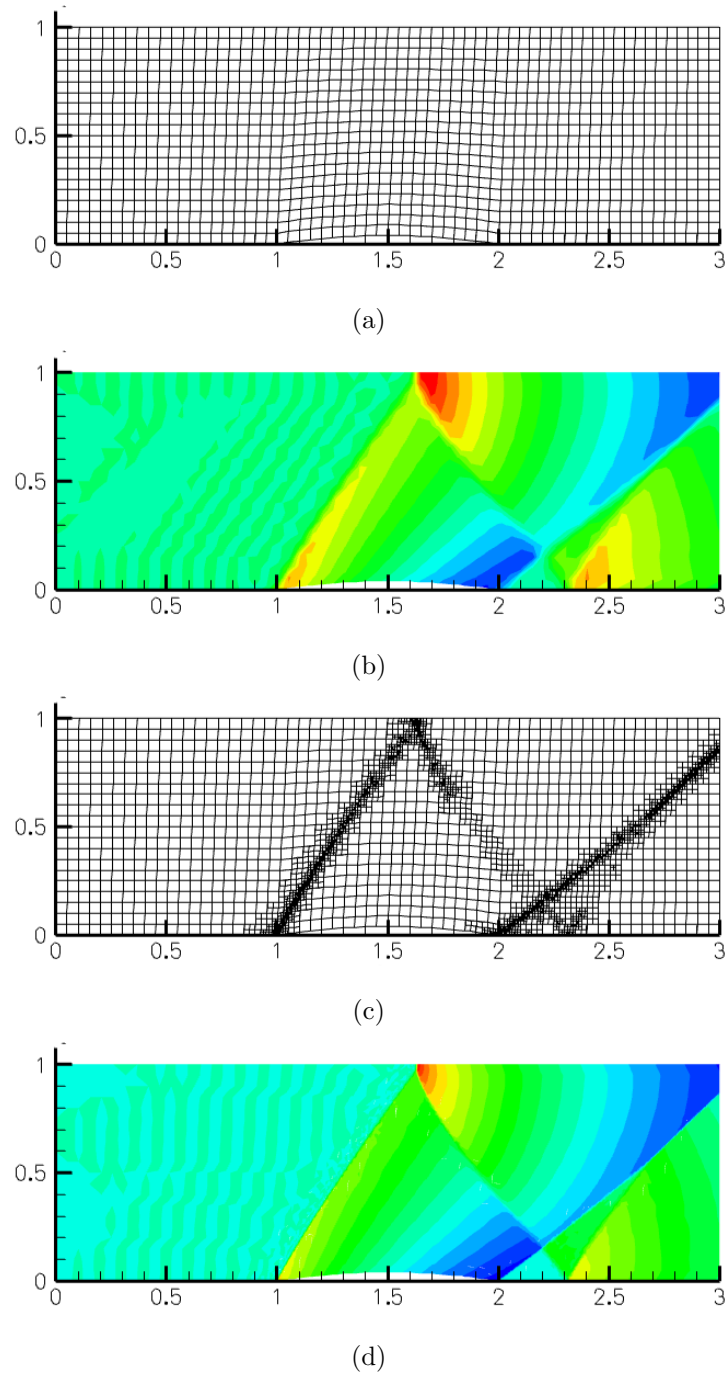


Figure 2: An example of how adaptive  $h$ -type mesh refinement can facilitate resolution of shock waves at Mach 1.4. The original mesh is shown in (a) and the original solution in (b). The automatically refined mesh is shown in (c) and the resulting refined solution in (d). Note that artificial viscosity is also employed to ensure that the SD scheme is stable in the presence of the shock waves.

### 2.3.5 Mesh Deformation

Mesh deformation algorithms have been implemented within our in-house 2D SD flow solver. In the schemes we have implemented, deformation of physical boundaries causes rigid displacement of nearby elements. This deformation is then blended smoothly into the mesh, such that the mesh at far field boundaries, or some other desirable portions of the flow domain, remains unaltered.

Deformation of the physical mesh is achieved via time-dependent variations of the metrics and the Jacobian that define the mapping of each physical element to a standard reference element. Such an approach preserves the high-order accuracy of the SD method since the governing equations are always solved in a steady reference element.

An example of mesh deformation and mesh blending is shown in Fig. 3. The undistorted passage of an Euler vortex across a mesh undergoing significant deformations is shown in Fig. 4.

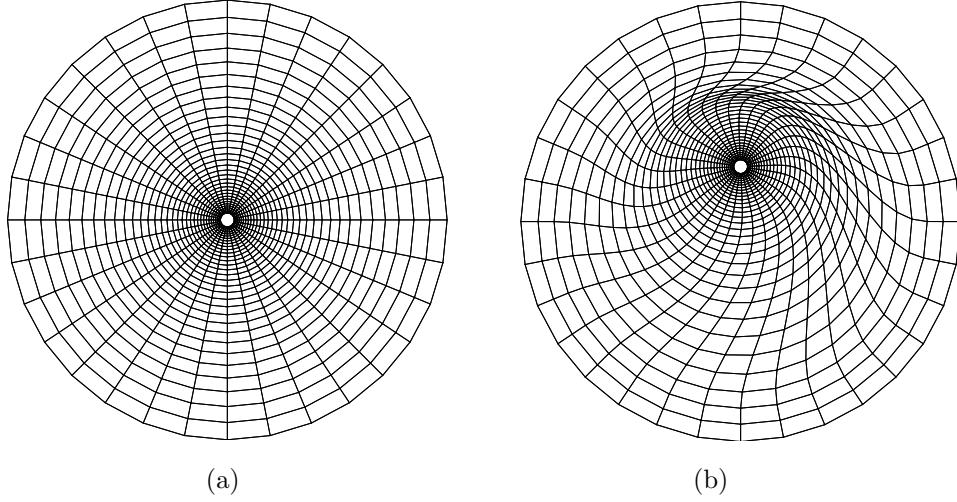


Figure 3: An illustration of how mesh deformation algorithms can be used to blend a rigid displacement of elements near a deformation into stationary elements at the boundary of the domain. The original mesh is shown in (a) and the resulting deformed mesh is shown in (b).

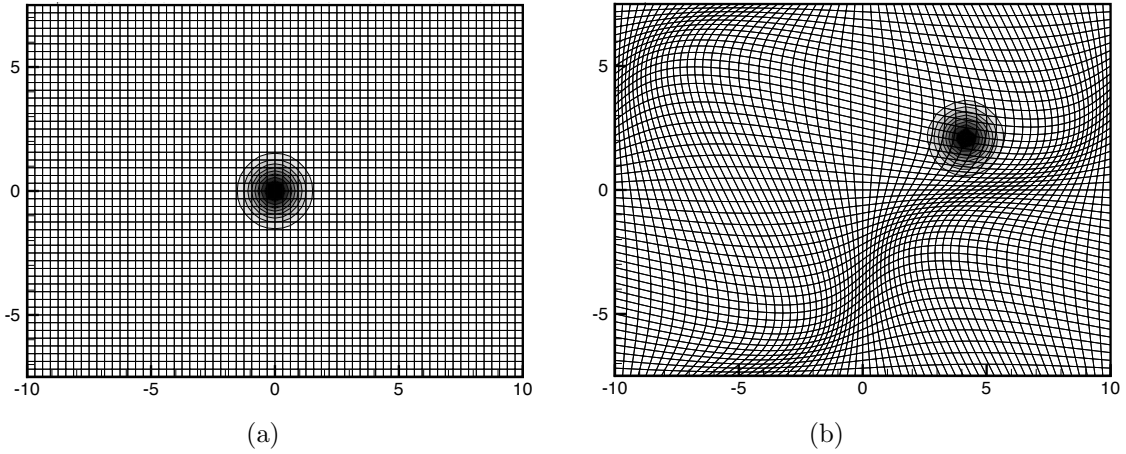


Figure 4: The initial pressure distribution in an Euler vortex is shown in (a). The pressure distribution after translation over a mesh undergoing spatial and temporal deformations is shown in (b). Note the the structure of the vortex remains uniform despite the non-uniform mesh deformation.

### 2.3.6 Convergence Acceleration

Efficient  $p$ -multigrid and implicit time integration methods have been implemented to accelerate the convergence of high-order SD schemes to steady-state [14]. To test the methods 2D steady viscous flow over an airfoil at  $Re = 5000$  has been investigated. A  $p$ -multigrid approach utilizing a 3-level V-cycle with 1-1-6-1-1 smoothing iterations was compared with an implicit time-stepping approach utilizing LU-SGS inner iterations. The convergence history of the residual is shown in Fig. 5. Results for an explicit third-order Runge-Kutta scheme are also shown for comparison. It can be seen that the  $p$ -multigrid approach reduced the computational cost by a factor of eight compared with the explicit third-order Runge-Kutta scheme, while the LU-SGS approach reduced the computational cost by factor of 100. These results indicate that by using efficient convergence acceleration techniques, the computational cost to reach a steady-state solution using the SD method can be greatly reduced.

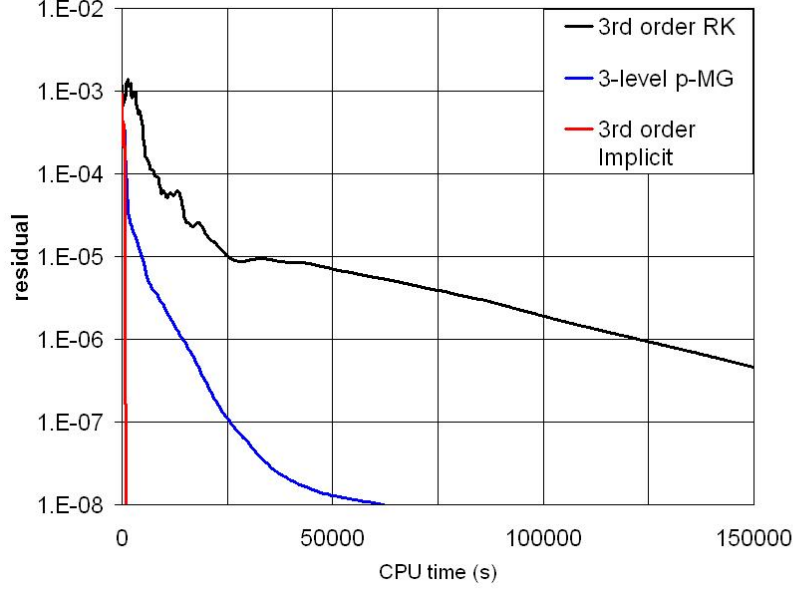


Figure 5: Plots of the residual against CPU time for an SD simulation of steady-state viscous flow over an airfoil at  $Re = 5000$ . Results for three time-integration schemes are shown. These include a  $p$ -multigrid approach utilizing a 3-level V-cycle with 1-1-6-1-1 smoothing iterations, an implicit time-stepping approach utilizing LU-SGS inner iterations, and an explicit third-order Runge-Kutta scheme. It can be seen that the  $p$ -multigrid approach reduces the computational cost by a factor of eight compared with the explicit third-order Runge-Kutta scheme, while the LU-SGS approach reduces the computational cost by factor of 100.

## 2.4 Code Utilization

### 2.4.1 Two-Dimensional Flow over Cylinders

Simulations of flow over a pair of stationary cylinders have been undertaken using our in-house 2D SD viscous compressible flow solver. Vorticity contours for cases where the cylinder separation is three times the cylinder diameter are shown for various Reynolds numbers ( $Re$ ) in Fig. 6. For further details see Liang, Premasuthan and

Jameson [15].

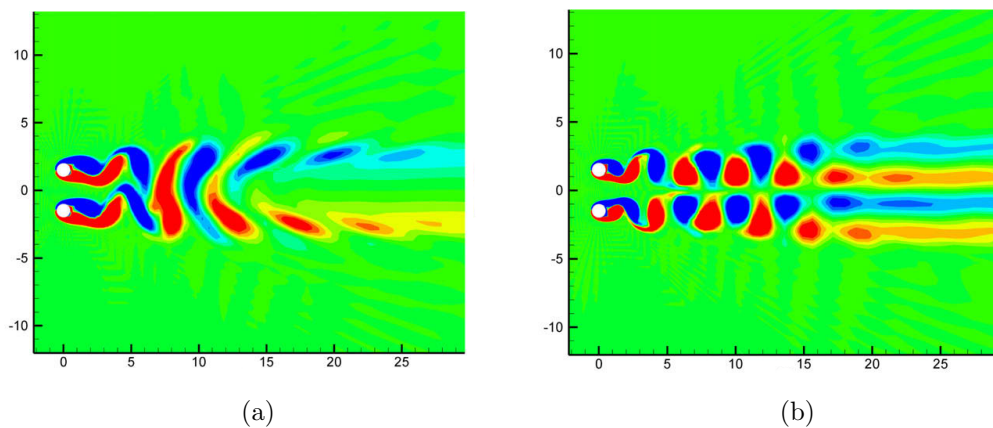
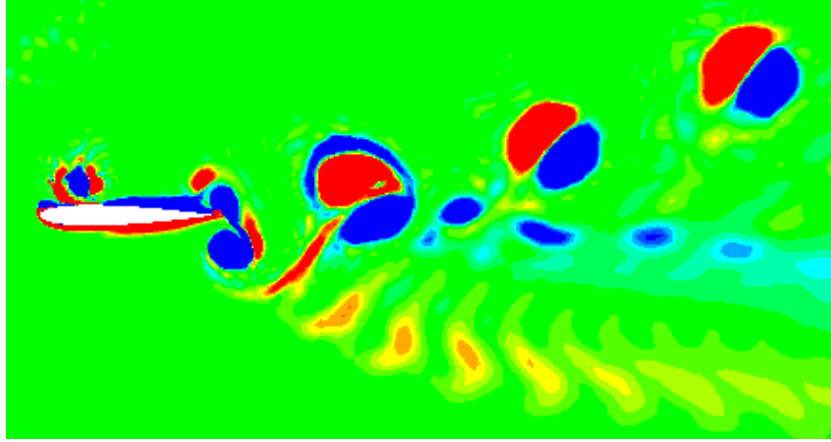


Figure 6: Plots of vorticity over a pair of stationary cylinders. Results are presented for  $Re = 100$  (a) and  $Re = 200$  (b).

Simulations of flow over a pair of counter rotating cylinders have also been undertaken using our in-house 2D SD viscous compressible flow solver. The effects of Reynolds number, compressibility, and rotation speed were all studied. For further details see Ou, Liang, Premasuthan and Jameson [16].

#### 2.4.2 Two-Dimensional Plunging and Pitching Airfoils

Simulations of flow over plunging and pitching NACA0012 airfoils have been undertaken using our in-house 2D SD viscous compressible flow solver. The plunging airfoil simulations were based on water tunnel experiments performed by Jones *et al.* [22] at  $Re = 1850$ . Fig. 7 shows that the pattern of vortical structures obtained via our numerical simulations compares well with experimental results. In particular, the simulations were able to reproduce the fine structures occurring in the wake of the plunging airfoil.



(a)

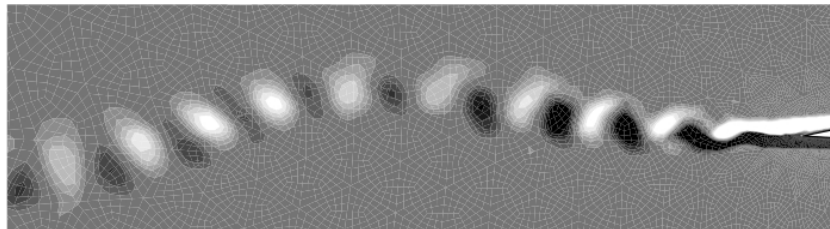


(b)

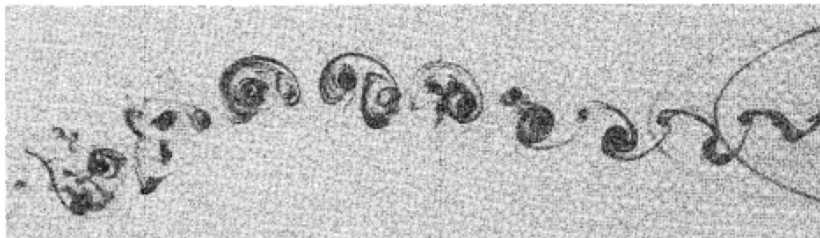
Figure 7: Vorticity over a plunging NACA0012 airfoil at  $Re = 1850$  calculated using a forth-order SD scheme (a) is compared with an analogous experimental result from Jones *et al.* [22] (b). The numerical results compare well with the experimental results. In particular, the simulations were able to reproduce the fine structures occurring in the wake of the plunging airfoil.

The pitching airfoil simulations were based on water tunnel experiments performed by Koochesfahani *et al.* [23]. Fig. 8 shows two comparisons between our numerical simulations and experiments (each with a different pitching frequency). In the first

case the wake assumes the form of an undulating vortex sheet, and in the second case a double-vortex structure is seen to persist downstream of the airfoil. In both cases results of our simulations are seen to compare well with the experimental data.



(a)

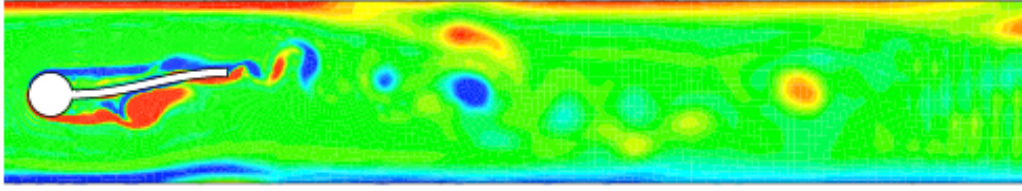


(b)

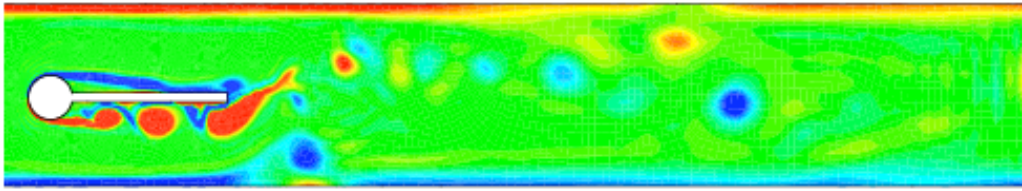
Figure 8: Vorticity over a pitching NACA0012 airfoil at  $Re = 1.2 \times 10^4$  calculated using a forth-order SD scheme (a) is compared with an analogous experimental result from Koochesfahani *et al.* [23] (b).

### 2.4.3 Two-Dimensional Interaction of a Cylinder Wake with a Deforming Beam

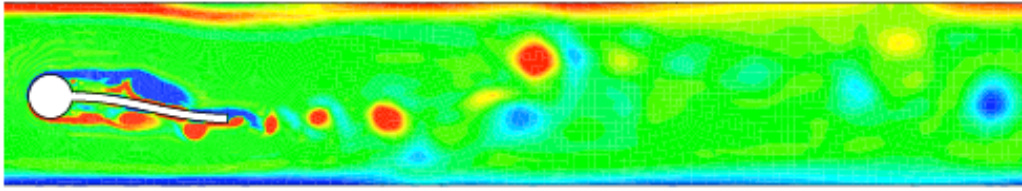
Simulations of flow over a cylinder connected to a beam undergoing a prescribed deformation have been undertaken using our in-house 2D SD viscous compressible flow solver. The setup acts as a simple 2D model of a body connected to a single flapping wing. Plots of vorticity within the vicinity of the configuration are shown at various instances in Fig. 9.



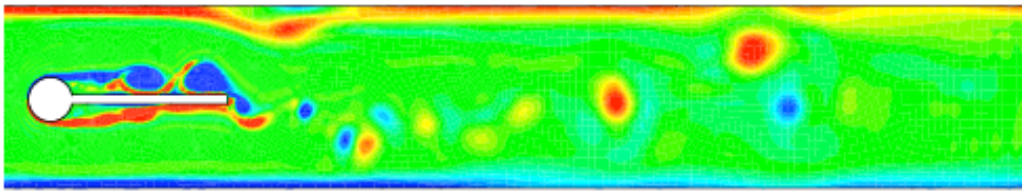
(a)



(b)



(c)



(d)

Figure 9: Plots of vorticity over a cylinder connected to a beam undergoing a prescribed deformation. Various instances in time are shown; specifically the peak of the upstroke (a), the following neutral position (b), the peak of the downstroke (c) and the following neutral position (d).

#### 2.4.4 Three-Dimensional Turbulent Channel Flow

Large Eddy Simulations of compressible turbulent channel flow were undertaken using the SD method [17]. Contours of spanwise vorticity in the center plane of the channel obtained using a fourth-order SD scheme are shown in Fig. 10. The predicted mean and root mean squared velocity profiles were found to be in good agreement with direct numerical simulation results obtained by Moser, Kim and Mansour [24].

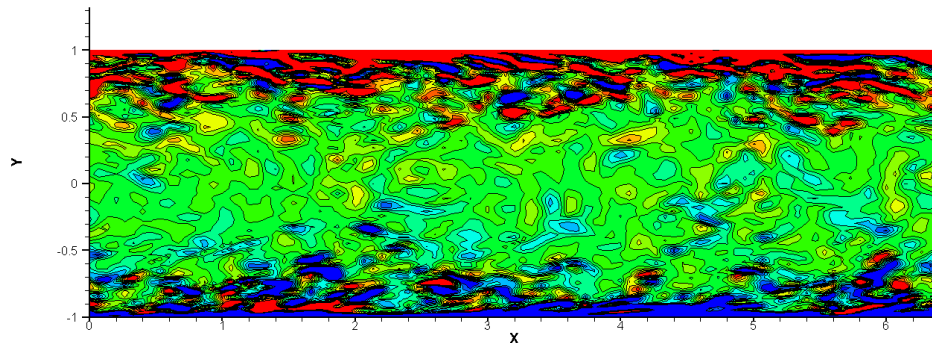


Figure 10: Contours of spanwise vorticity in the center plane of the channel obtained using a fourth-order SD scheme.

#### 2.4.5 Three-Dimensional Turbulent Flow Over an Airfoil

Simulations of flow over an SD7003 airfoil at a  $4^\circ$  angle of attack have been performed using our in-house 3D viscous compressible SD flow solver. The SD7003 airfoil was selected due to availability of existing experimental [25] and computational data [26]. Forth-order accurate simulations were undertaken on meshes with  $1.7 \times 10^6$  degrees of freedom at  $Re = 1 \times 10^4$  and  $Re = 6 \times 10^4$ . When  $Re = 1 \times 10^4$  the flow was essentially 2D with close-to-periodic vortex shedding (see Fig. 11(a)). The computed

average lift coefficient was 0.381 and the average drag coefficient was 0.0497, which are consistent with those obtained by Uranga *et al.* [26]. At a Reynolds number of  $Re = 6 \times 10^4$  transition was observed to take place across a laminar separation bubble (see Fig. 11(b)).

The Q-criterion  $Q$  provides a mean of visualizing vortex cores and identify turbulent structures. It can be calculated as

$$Q = \frac{1}{2}(\Omega_{ij}\Omega_{ij} - S_{ij}S_{ij}) \quad (2.2)$$

where

$$\Omega_{ij} = \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right), \quad S_{ij} = \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (2.3)$$

are the anti-symmetric and symmetric parts of the velocity gradient tensor respectively. Fig. 11 shows instantaneous iso-surfaces of  $Q$  over the SD7003 airfoil for cases where  $Re = 1 \times 10^4$  and  $Re = 6 \times 10^4$ .

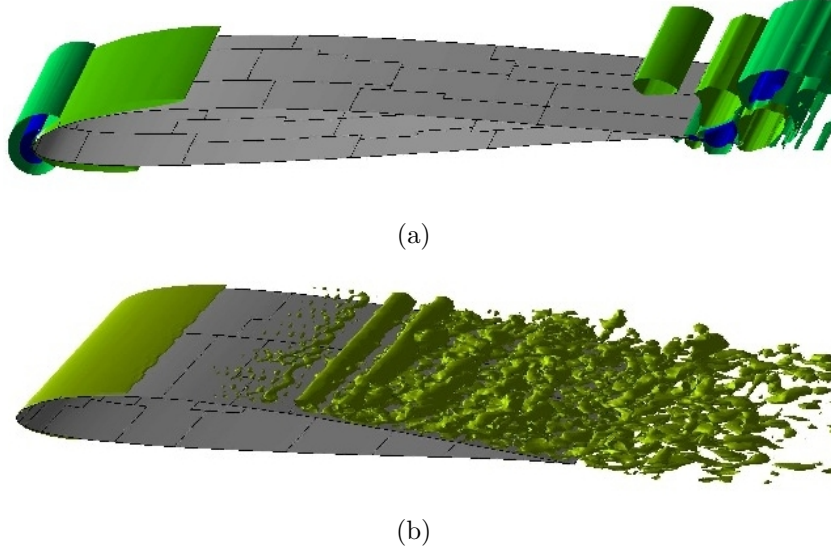


Figure 11: Instantaneous iso-surfaces of  $Q$  above an SD7003 airfoil at a  $4^\circ$  angle of attack. When  $Re = 1 \times 10^4$  (a) the flow is essentially 2D and remains laminar over the wing surface, while at  $Re = 6 \times 10^4$  (b) transition takes place across a laminar separation bubble.

Use of the SD method with an implicit large eddy simulation approach appears to be capable of accurately predicting the laminar separation and transition locations over an SD7003 airfoil at  $Re = 6 \times 10^4$ . To the best of our knowledge, this study is the first attempt to analyze transitional flow using the SD method.

#### 2.4.6 Three-Dimensional Flapping Wing

Preliminary simulations of viscous compressible flow over a SD7003 airfoil undergoing a prescribed flapping motion have been undertaken using our in-house 3D viscous compressible SD flow solver. The geometry and nature of the prescribed motion are illustrated in Fig. 12.

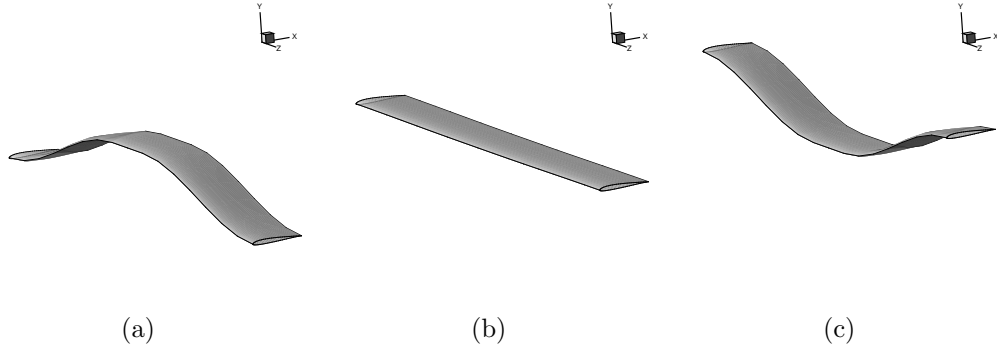


Figure 12: Deforming SD7003 airfoil at bottom of cycle (a), middle of cycle (b), and top of cycle (c).

### 3 Conclusions

A summary has been given of research carried out under award number FA9550-07-1-0195 from the Air Force Office of Scientific Research. The objective of the proposed research was to develop and implement high-order numerical algorithms for the simulation of steady and unsteady compressible viscous flows with shocks. Notable achievements have been made in three areas, namely algorithm analysis and development, code development, and code utilization (to investigate various flow problems). With regards to algorithm analysis and development, it has been proved for one-dimensional linear advection that the SD method is stable for all orders of accuracy in a norm of Sobolev type (provided that the interior flux collocation points are placed at the zeros of the corresponding Legendre polynomials). Also, a new range of energy stable high-order methods based on the so called FR approach have been identified. With regards to code development, two-dimensional and three-dimensional compressible viscous flow solvers based on the spectral difference method have been

written. Within the aforementioned codes a range of shock capturing, automatic mesh refinement, mesh deformation, and convergence acceleration algorithms have been implemented and tested. With regards to code utilization, viscous compressible flow over various 2D configurations has been investigated. These configurations include pairs of cylinders (both stationary and rotating), plunging and pitching airfoils, and a deforming beam in the wake of a cylinder. In 3D, turbulent channel flow and turbulent flow over an airfoil have been investigated, and preliminary simulations of viscous compressible flow over a flapping wing have been undertaken.

## References

- [1] W. H. Reed and T. R. Hill, *Triangular mesh methods for the neutron transport equation*, Technical Report LA-UR-73-479, Los Alamos National Laboratory, Los Alamos, New Mexico, USA, 1973.
- [2] B. Cockburn and C. Shu, *Runge–Kutta Discontinuous Galerkin Methods for Convection-Dominated Problems*, J. Sci. Comput. **16**, 173 (2001).
- [3] D. N. Arnold, F. Brezzi, B. Cockburn, and L. D. Marini, *Unified analysis of discontinuous Galerkin methods for elliptic problems*, SIAM J. Numer. Anal. **39**, 1749 (2001).
- [4] J. S. Hesthaven and T. Warburton, *Nodal discontinuous Galerkin methods - Algorithms, analysis, and applications*, Springer, (2008).
- [5] J. S. Hesthaven and T. Warburton, *Nodal high-order methods on unstructured grids*, J. Comput. Phys. **181**, 186 (2002).
- [6] F. X. Giraldo, J. S. Hesthaven, and T. Warburton, *Nodal high-order discontinuous Galerkin methods for the spherical shallow water equations*, J. Comput. Phys. **181**, 499 (2002).
- [7] D. A. Kopriva and J. H. Kolas, *A conservative staggered-grid Chebyshev multidomain method for compressible flows*, J. Comput. Phys. **125**, 244 (1996).
- [8] Y. Liu, M. Vinokur, and Z. J. Wang, *Spectral difference method for unstructured grids I: basic formulation*, J. Comput. Phys. **216**, 780 (2006).

- [9] A. Jameson, *A proof of the stability of the spectral difference method for all orders of accuracy*, J. Sci. Comput. (2010).
- [10] P. E. Vincent, P. Castonguay, and A. Jameson, *A New Class of High-Order Energy Stable Flux Reconstruction Schemes*, J. Sci. Comput. (Submitted).
- [11] S. Premasuthan, C. Liang, and A. Jameson, *A spectral difference method for viscous compressible flows with shocks*, AIAA Computational Fluid Dynamics Meeting (2009).
- [12] S. Premasuthan, C. Liang, and A. Jameson, *Computation of flows with shocks using spectral difference scheme with artificial viscosity*, AIAA Aerospace Sciences Meeting (2010).
- [13] K. Ou, C. Liang, and A. Jameson, *A high-order spectral difference method for the Navier-Stokes equations on unstructured moving deformable grids*, AIAA Aerospace Sciences Meeting (2010).
- [14] S. Premasuthan, C. Liang, A. Jameson, and Z. J. Wang, *A p-multigrid spectral difference method for viscous compressible flow using 2D quadrilateral meshes*, AIAA Aerospace Sciences Meeting (2009).
- [15] C. Liang, S. Premasuthan, and A. Jameson, *High-order accurate simulation of low-Mach laminar flow past two side-by-side cylinders using spectral difference method*, Comput. Struct. **87**, 812 (2009).
- [16] K. Ou, C. Liang, S. Premasuthan, and A. Jameson, *High-order spectral difference simulation of laminar compressible flow over two counter-rotating cylinders*, AIAA Computational Fluid Dynamics Meeting (2009).

- [17] C. Liang, S. Premasathan, A. Jameson, and Z. J. Wang, *Large eddy simulation of compressible turbulent channel flow with spectral difference method*, AIAA Aerospace Sciences Meeting (2009).
- [18] H. T. Huynh, *A flux reconstruction approach to high-order schemes including discontinuous Galerkin methods*, AIAA Computational Fluid Dynamics Meeting (2007).
- [19] Y. Sun, Z. J. Wang, and Y. Liu, *High-order multidomain spectral difference method for the Navier-Stokes equations on unstructured hexahedral grids*, Commun. Comput. Phys. **2**, 310 (2007).
- [20] C. Liang, A. Jameson, and Z. J. Wang, *Spectral difference method for compressible flow on unstructured grids with mixed elements*, J. Comput. Phys. **228**, 2847 (2009).
- [21] A. W. Cook and W. H. Cabot, *Hyperviscosity for shock-turbulence interactions*, J. Comp. Phys. **203**, 379 (2005).
- [22] K. D. Jones, C. M. Dohring, and M. F. Platzer, *Experimental and computational investigation of the Knoller–Betz effect*, AIAA Journal **36**, 780 (1998).
- [23] M. M. Koochesfahani, *Vortical patterns in the wake of an oscillating airfoil*, AIAA Journal **27**, 200 (1989).
- [24] D. Moser, J. Kim, and N. N. Mansour, *Direct numerical simulation of turbulent channel flow up to  $Re=590$* , Phys. Fluids **11**, 943 (1999).

- [25] R. Radespiel, J. Windte, and U. Scholz, *Numerical and experimental flow analysis of moving airfoils with laminar separation bubbles*, AIAA Journal **45**, 1346 (2007).
- [26] A. Uranga, P. Persson, M. Drela, and J. Peraire, *Implicit large eddy simulation of transitional flows Over airfoils and wings*, Technical report, Massachusetts Institute of Technology, 2009.